Journal of Advances in Mechanical Engineering and Science, Vol. 1(1) 2015, pp. 20-33

RESEARCH ARTICLE

Combustor Characteristics under Dynamic Condition during Fuel – Air Mixing using Computational Fluid Dynamics

Md. Shamim Raza[1], Nitesh Kumar[2], *Sourav Poddar[3]

[1]Dept. of Mechanical Engineering, Ram Govind Institute of Technology, Koderma, Jharkhand, India

[2]Dept. of Mechanical Engineering, Jadavpur University, Kolkata, India

[3]Dept. of Chemical Technology, University of Calcutta, Kolkata, India

Received-21 May 2015, Revised-26 June 2015, Accepted-8 August 2015, Published-8 August 2015

ABSTRACT

A gas turbine can combustor is designed to burn the fuel efficiently while reducing the NOx and CO emissions, and lowering the wall temperature. Environmental challenges with gas turbine include low levels of NOx, CO and soot amongst other pollutants. There is a need for new concepts and technology to satisfy the pollutants emission regulations and to enhance energy conservation. Specifically, ultra-low NOx combustor technology is required to meet the ozone depletion challenge. Researchers now face a challenge of developing dry low-NOx emitting stationary and aero engines. However, any concept for environmental pollution control requires a detailed understanding of the physical and the chemical processes that occur during combustion. In this paper, a three dimensional numerical investigation of the Combustion methane air mixture in a gas turbine can combustor is carried out by using ANSYS. The objective of the study is to understand the combustion phenomena at different planes. The various parameters like air-fuel ratio, velocity of primary air inlet are used to investigate the effects on parameters like combustion chamber on different plane performance and emission. A premixing tube is augmented with the combustion chamber which has a primary air inlet port and three gaseous fuel inlet ports. Air–methane mixture is considered to enter the combustion zone with inlet swirl. The homogeneity of mixture before and after swirl and other important conditions are calculated from simulation and are reported with the help ANSYS FLUENT. Weighted averages of velocity magnitude distribution and mass fractions of methane have been studied at different planes.

Keywords: Air-methane mixture, ANSYS FLUENT, Combustion chamber, Gas turbine, Can combustor.

1. INTRODUCTION

1.1. Gas Turbine

A Gas Turbine is a heat engine which uses fuel energy to produce mechanical output power, either as torque through a rotating shaft (industrial gas turbines) or as jet power in the form of velocity through an exhaust nozzle (aircraft jet engines), thus producing electrical energy from burning a combustible mixture of fuel (e.g. natural gas or evaporated hydrocarbons) and air. When the gas mixture burns, the volume of the gas increases. This expansion in gas volume makes the rotor of the turbine to rotate and this rotation is then converted into electrical energy. The schematic diagram of a gas turbine is shown in figure A1. There are two important families of gas turbines:

1. Stationary gas turbines: These type of turbines are used to produce power in large scales, for example in power plants.
2. Turbofan and turbojet gas turbines: These turbines are usually used as aircraft engines and are sometimes referred to as jet engines. A variety of turbofan and turbojet gas turbines are used in military and commercial aircrafts.

*Corresponding author. Tel.: +919230550785

Email address: cupidsourav@gmail.com (S.Poddar)

Double blind peer review under responsibility of DJ Publications

http://dx.doi.org/10.18831/james.in/2015011003

1.2. Combustion chamber

The combustion chamber is the place where two major events take place; one is the mixing of air and fuel and the second is the actual combustion. Depending upon whether the mixing of air and fuel is done before the combustion or during the combustion, combustors are divided into two groups namely diffusion flame combustors and premixed flame combustors. A brief explanation about the two types of combustors is given below.

- Diffusion Flame Combustors: In diffusion flame combustors, fuel and air mixing and combustion takes place simultaneously. Speed of flame is limited by the rate of diffusion. These kinds of combustors are simple to build and operate, but they are not environmentally clean. The major drawback of these combustors is that the flame exists mainly at stochiometric conditions. This can result in high rates of NOx production.
- Premixed flame combustors*:* These combustors are newer than the diffusion flame combustors. They mix the fuel with air to a high degree thus the flame exists where the fuel exists, if it can be stabilized. Contrary to the previous combustor type these combustors are more complex and harder to design, but they produce less NOx. Every day new challenges arise for gas turbines. Different factors like increase in oil price, new type of fuels like bio fuels, different designs like premix combustors and many other factors will challenge engineers to develop new combustors or improve the existing combustors.

However, in order to achieve a smooth burning, air and fuel should be mixed before burning. During the combustion the fuel-air mixture ignites and raises the temperature. Rise in temperature increases the volume which in turn drives the fluid forward. The design of the combustion chamber plays a vital role in controlling the combustion process. Some important problems occurring as a result of poor design of a combustion chamber are listed below:

1) Poor mixing: When fuel is not mixed enough with air, it can burn incompletely which results in increased levels of CO, soot, NOx and unburned hydrocarbons (UHC)
2) Uneven combustion: This happens when temperature of a section goes high while the neighboring sections are colder, which can result in extra thermal stresses. At times thermal stresses may lead to material fatigue and failure.
3) Environment: Incompletely burned gases or unburned hydrocarbons (UHC) can poison the environment. UHC, NOx and soot are important factors for each burning device. The design should lower them as much as possible.
4) Economy: With increasing price of oil, it is important that gas turbines have high efficiency and therefore low fuel consumption. One of the most important parts, in order to achieve high efficiency, is the combustion chamber.

1.3. Previous work in the area

The development of advanced combustion capability for gaseous hydrogen and hydrogen-blended hydrocarbon fuels in gas turbine application is an area of much interest. In most countries NOx emissions of stationary gas turbines are regulated strictly. To satisfy these regulations, modern gas turbine manufacturers rely mostly on lean pre-mixed combustion. Mixing large amounts of air with gas prior to burning reduces the flame peak temperature and thereby leads to lower NOx and CO emissions. Efficient combustion is the key integral part not only in power generation related sectors like gas turbine, aircraft engines etc. but also in industrial burners where in diffusion flame has been in use for a long time.

In the present study, emphasis is placed on the fuel preparation process which is an important factor and sometimes play a key role in the reduction of pollutant emission from gas turbines [1]. It is a challenge to handle the environmental problems, with low pollution from a broad range of fuels [2]. New concepts introduced to gas turbine industry include lean premixed combustion (LPM), rich-burn quick-Quench lean-burn (RQL) combustion, catalytic combustion and selective catalytic reduction (SQR); high - efficiency low emissions

are being extended to non-premium fuels such as coal gas and crude oil; new materials such as super alloys, thermal barrier coatings, and ceramics have been incorporated into designs; and improved theories are greatly dependent on advanced laser-based diagnostics of flame structure and have led to design tools of increasing scope [3]. Among these, the lean premised combustion appears to be the most effective mainly because of its two-fold advantages [4] and [5] i.e., Reduction of flame temperature owing to the presence of excess air which eliminates production of thermal NOx and low exhaust temperature thereby increasing the life time of turbine blades and other mechanical components.

The primary drawback of such LPM combustors is reduction in the operability range between the flashback and blowout regime which decreases the overall stability of the combustion process [6]. Flame stabilization schemes that have been widely used in such applications mainly involve bluff body or swirl stabilization or a combination of both. The physical process that anchors a premixed flame behind a bluff-body flame holder involves mixing of the combustible mixture, which are residing in the recirculation zone. The popular stabilization mechanisms used for sustaining the flame in the combustor are bluff body flame holders which is reported by the previous researchers [7] and are taken into account by the present research group. In this process bluff-body flame holder involves in mixing of the combustible mixture with the hot combustion products residing in the recirculation zone, thus facilitating continuous ignition of the premixed reactants in its vicinity are considered and used in the process used by the present research group as a parameter. The popular stabilization mechanisms used for sustaining the flame in the combustor are bluff body flame holders [8]. Dump plane swirler and pilot flame Lean blowout (LBO) is a very important aspect of combustion performance for gas turbine combustor. The problem of LBO was always regarded as relatively minor. This was to a large extent, due to the almost universal use of pressure-swirl spray atomizers. The poor mixing characteristic of these pressure swirl atomizer creates many performance problems, the last being a high rate of soot formation in the primary zone. However, this poor mixing of fuel and primary air has the useful advantage of allowing combustion to occur. In recent years, the trend towards improving fuel/air mixture prior to combustion in order to reduce pollutant emission has led to a narrowing of stability limits and increasing the satisfactory LBO performance. A failure in sustaining the flame in the combustors lead to the phenomenon called lean blowout (LBO). To avoid such unwanted phenomena, current combustion are typically operated with a wide margin above the lean blowout limit which tends to increase the NOx formation from the engine. Hence, a significant optimized design, balancing operation economy and reduction of emission is crucial. The current work is aimed at addressing a part of this issue where a CFD analysis is carried out on dump plane swirler geometry to investigate the homogeneity of air-fuel mixture [9]. A complete active control system sensing, actuation, and control algorithm has been developed to permit turbine engine combustion to operate safely closer to the lean-blowout (LBO) limit, even in the presence of disturbances. The system use OH chemiluminescence. Chemiluminescence is directly related to chemical reaction rates and it can provide information on the presence and strength of the combustion process in a specific region of the combustor [10]. Though plenty of sensing tragedies for predicting the proximity of combustion to lean blowout have been reported in the past, the general trend is to study the flame dynamic near the blowout limit at a fixed degree of premixedness. The degree of premixedness influence heat release fluctuations which being the primary cause of dynamic instability and subsequently affect the LBO limit [11]. Several experimental studies characterizing the blowout phenomenon along with secondary measurement like strain rate at flame base pulse rate before the onset of lift -off have been done using modern optical diagnostic tools like LIPF(Laser-induced Predissociative Fluorescence), PIV(Particle-Image Velocity) and OH PLIF(Planar Laser Induced Fluorescence) [12]. Optical OH chemiluminescence has also been used by [13] to observe extinction and re-ignition events near the blowout event which has been called as precursor events. LBO prediction is a very important aspect in combustion scenario and various methods have been established for the same over the years. The very primitive of such methods was based on monitoring pressure in the combustion chamber.

1.4. Scope of present work

The focus of the present work is to study the dynamic characteristics of fuel-air mixing in a model gas turbine dump combustor. The model geometry is shown in Fig. 2, where air is entering at one end of the combustor and gaseous fuel is entering from one of the perpendicular ports shown (port-1). The air velocity is varied sinusoidally with time and the effect of this mixing parameter is studied at different planes. The work is carried out numerically using FLUENT software.

2. CALCULATION METHODOLOGY

2.1. Problem description

The combustion chamber, 3D geometry, is generated using ANSYS FLUENT. The combustion zone is cylindrical in shape with 20cm length and 6cm internal diameter. Before the combustion zone a premixing tube is provided with a maximum premixing length of 35cm and diameter of 2.3 cm. Dump plane is considered as the plane at which the combustion zone starts with an abrupt increase of cross-section. At the bottom of the premixing tube i.e. 37.5 cm from the dump plane the air inlet port of 1.5cm diameter is provided. There are 3 fuel inlet ports each of 6mm diameter at 10 cm distance from each other, the uppermost being 15cm below the dump plane. The fuel inlet ports which are at a distance of 35cm, 25cm and 15cm from the dump plane are named as port 1, port 2 and port 3 respectively. Just below the dump plane a swirler of length 1.5 cm and blade angle with axial direction 60° is placed. The number of blades on the swirler is 8 and blade thickness is 1mm. Diameter of the swirler section is 2.9cm. Mesh generated is unstructured and in standard form as provided by ANSYS. The number of nodes and elements are 3371and12346 respectively. Details may be found in figure A2.

For Numerical simulation of the generated meshed geometry in 3D, pressure-based, transient, species transport and k-∈ turbulent models are utilized. For species transport model methane-air mixture is used for all cases. For fuel (Methane), entry through port-1 the air flow rate of 85 litres per minute (LPM) are used, the corresponding air inlet velocity being 8.0208m/s. For entry of fuel (Methane) through port -1, the premixing length is 35 cm. The fuel flow rate is varied and equivalent ratio and fuel entry port is considered as 1 for each case respectively. For this equivalent ratio the fuel inlet velocity is maintained at 1.062m/s. The most common combustors have separate inlets for fuel and air. Both the fuel and the air are assumed to be perfectly mixed before combustion chamber. Homogeneity in air-fuel mixture with varying premixing length with respect to contours of mass fraction of methane (CH_4) and velocity magnitude at different important planes inside and outside the combustor is found. Planes are generated at a distance of 3.5cm before the dump plane and 15cm, and 20 cm after dump plane.FigureA3 and figureA4 show the graphs and contour constructed to display the distribution of mass fraction of methane and velocity magnitude along a time. Results are presented for most important comparisons and variations with time.

2.2. Boundary conditions

Boundary conditions are as follows

1) Air Inlet port: UDF Unsteady Velocity,
2) Turbulence: Specification Method – Intensity and Hydraulic Diameter.
   Turbulent intensity -10%
   Hydraulic Diameter-0.015m
3) Port-1 : Velocity 1.062(m/s)
   Turbulence: Specification Method – Intensity and
   Hydraulic Diameter. Turbulent intensity -10%
   Hydraulic Diameter-0.06m
4) Walls: Adiabatic and free slip
5) Reference pressure: 0 (Pascal)

2.3. Methodology

A commercial available CFD code FLUENT has been used for the analysis. Five equations are used to model the flow field. These equations are continuity, momentum, swirler velocity, species transport and turbulence. The following sections will describe these equations.

2.4. UDF equation

The UDF equation is shown in equation (2.1)

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{v}) = 0 \quad (2.1)$$

2.5. Continuity equation or mass conservation equation

This equation describes that what comes in should go out. Mass can neither be destroyed nor created.

The general form of the mass conservation equation is given by the e following equation

$$\frac{\partial(\rho v)}{\partial t} + \nabla.(\rho v v) = -\nabla_{\mathrm{p}} + \nabla.(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \quad (2.2)$$

Equation (2.2) is and is valid for incompressible as well as compressible flows.

2.6. Momentum conservation equation

Momentum is a vector quantity that is the product of mass by velocity vector. In a closed system, momentum cannot be created nor destroyed. It should be conserved. The equation (2.3) reads,

$$\bar{\bar{\boldsymbol{\tau}}} = \mu[(\nabla\vec{v} + \nabla\vec{v}^{T}) - \frac{2}{3}\nabla.\vec{v}I] \quad (2.3)$$

Where p is the static pressure, $\boldsymbol{\tau}$ is the stress tensor (described below), and $\boldsymbol{\rho g}$ and $\boldsymbol{F}$ are the gravitational body force and external body forces, respectively. $\boldsymbol{F}$ also contains other model-dependent source terms such as user-defined sources. The stress tensor $\boldsymbol{\tau}$ is given by equation (2.4).

$$\frac{\partial}{\partial t}(\rho w) + \frac{1}{r}\frac{\partial}{\partial x}(r\rho u w) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho v w) = \frac{1}{r}\frac{\partial}{\partial x}[r\mu\frac{\partial w}{\partial x}] + \frac{1}{r^2}\frac{\partial}{\partial r}[r^3\mu\frac{\partial}{\partial r}(\frac{w}{r})] - \frac{\rho v w}{r}. \quad (2.4)$$

2.7. Momentum conservation equation for swirl velocity

The tangential momentum equation (2.5) for 2D swirling flows may be written as

$$\frac{\partial}{\partial t}(\rho y_i) + \nabla.(\rho\vec{v}y_i) = -\nabla.\vec{J_i} + R_i + S_i \quad (2.5)$$

2.8. Species transport equations

When you choose to solve conservation equations for chemical species, ANSYS FLUENT predicts the local mass fraction of each species, Yi, through the solution of a convection diffusion equation(2.6) for the i_{th} species.

$$\frac{\partial}{\partial t}(\rho y_i) + \nabla.(\rho\vec{v}y_i) = -\nabla.\vec{J_i} + R_i + S_i \quad (2.6)$$

Where Ri is the net rate of production of species, i be the chemical reaction and Si is the rate of creation by addition from the dispersed phase plus any user-defined sources. In this equation (2.6) we assume that, Si is equal to zero. An equation of this form will be solved for N −1 species where N is the total number of fluid phase chemical species present in the system. Since the mass fraction of the species must sum to unity, the Nth mass fraction is determined as one minus the sum of the N − 1 solved mass fractions. To minimize numerical error, the Nth species should be selected as that species with the overall largest mass fraction, such as N2 when the oxidizer is air.

2.9. Turbulence models

2.9.1. Mass diffusion in turbulent flows

In turbulent flows, ANSYS FLUENT computes the mass diffusion equation (2.7) in the following form,

$$\vec{J_i} = -\left(\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right)\nabla y_i - D_{T,i}\frac{\nabla \mathrm{T}}{\mathrm{T}} \quad (2.7)$$

Where Sct is the turbulent Schmidt number, μ_t is the turbulent viscosity and D_T is the turbulent diffusivity. The default is Sct 0.7.

2.9.2. k-$\boldsymbol{\varepsilon}$ Turbulence model

The k-$\boldsymbol{\varepsilon}$ model is one of the most common turbulence models. It is a two equation model which means that two extra transport equation is included to represent the turbulent properties of the flow. In these cases, using mixture properties and mixture velocities is sufficient to capture important features of the turbulent flow. The k and $\boldsymbol{\varepsilon}$ equations (2.8) and (2.9) describing this model are as follows

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla.(\rho_m\overrightarrow{v_m}k) = \nabla.\left(\frac{\mu_{t,m}}{\sigma_k}\nabla k\right) + G_{k,m} - \rho_m\epsilon \quad (2.8)$$

$$\frac{\partial}{\partial t}(\rho_m\epsilon) + \nabla.(\rho_m\overrightarrow{v_m}\epsilon) = \nabla.\left(\frac{\mu_{t,m}}{\sigma_\epsilon}\nabla\epsilon\right) + \frac{\epsilon}{k}(C_{1\epsilon}G_{k,m} - C_{2\epsilon}\rho_m\epsilon) \quad (2.9)$$

where the mixture density and velocity, $\rho_m$and vm are computed from eqautions(2.10) and (2.11)

$$\rho_m = \sum_{i=1}^{N}\alpha_i\,\rho_i \quad (2.10)$$

and

$$\overrightarrow{v_m} = \frac{\sum_{i=1}^{N}\alpha_i\rho_i\vec{v}_i}{\sum_{i=1}^{N}\alpha_i\rho_i} \quad (2.11)$$

The turbulent viscosity, m is computed from equation (2.12)

$$\mu_{t,m} = \rho_m C_\mu\frac{K^2}{\epsilon} \quad (2.12)$$

and the production of turbulence kinetic energy, $G_{k,m}$, is computed from equation(2.13)

$$G_{k,m} = \mu_{t,m}\{\nabla\overrightarrow{v_m} + (\nabla\overrightarrow{v_m})^T\}:\nabla\overrightarrow{v_m} \quad (2.13)$$

3. RESULTS AND DISCUSSIONS

The area weighted average of methane, mass fraction of CH_4, area weighted average velocity magnitude, mass weighted average of methane are calculated. Figure 5 shows the Mass weighted Average Velocity magnitude contour plots, where it is found that the result

for dump plane, dump plane attached with swirler and the plane created are 3.5cm, 15 cm and 20 cm respectively. For air flow, velocity of UDF [where uo =8.062, A = 4% of uo w = 95] at air inlet port (port -1) having a constant velocity is 1.062m/s. While Fig. 8 shows mass weighted average velocity magnitude distribution along the above mentioned planes with respect to time.

The scale utilized in important planes before and after dump plane is 0 to 5m/s which is significant with respect to the Velocity Magnitude. For the planes constructed before and after dump plane the same scale is used and methane mass fraction of CH_4 scales 0 to 0.023.

3.1. Velocity magnitude

Homogeneity in air-fuel mixture is obtained by using the UDF Velocity at air inlet port and constant velocity at port-1.The variation of velocity magnitude with respect to distance and time in different important plane before swirler and after swirler is obtained. We study from the contour at YZ plane (figureA5) and graph (figureA7) that the velocity magnitude is very large before swirler and after swirler. Attached with the swirler is called dump plane. Another two important plane in a combustion chamber is 15 cm after dump plane and 20cm after dump plane in which the velocity decreases gradually with distance as well as time as the flow progresses through the swirler(magnitude of velocity before swirler plane is 3.5 m/sec. and after swirler dump plane it is 0.56 m/sec,,15cm after dump plane it is 0.494 m/sec,,20cm after dump plane it is 0.479 m/sec) .After swirler, flow enters into the combustion chamber inlet that is called the dump plane where velocity further reduces due to large diameter difference between before swirler plane (φ2.3cm)and after swirler plane(φ6cm) resulting in the formation of wall recirculation zone (figureA6). The plane diameter in a combustion zone is more than double with respect to premixed tube diameter and swirlers also play a measure role to oppose the velocity of Air-fuel mixture.

3.2. CH_4 mass fraction

In order to achieve better mixing between fuel and air in combustor, turbulent flow must be generated to promote mixing. Below figure shows the CH_4 mass fraction contours at YZ planes while figureA9(a), shows CH_4 mass fraction contours at plane before swirler compared with dump plane and other two different important planes after swirler .FigureA9 is important as they show homogeneity of air -fuel mixture and mixture pattern at different position along the length of the combustor. Comparing (a), (b), (c), (d) of figureA9, it can be easily detected that in the mixture before swirler and after swirler in the combustor the methane mass fraction regularly decreases at different plane with increase in distance and time. FigureA10 shows a plot of mass fraction of CH4 versus time for different planes. There was decreasing fluctuation of mass fraction of CH4 while increasing the distance. The fluctuation difference between maximum and minimum methane mass fraction values are 0.0006, 0.0001, 0.000022, 0.0000038 for plane position 3.5cm before dump plane, dump plane, 15cm after dump plane, 20cm after dump plane respectively. This shows more rapid decrease of the difference for plane 20cm after dump plane when compared to 3.5 cm before dump plane. This means that fluctuation is almost steady on the plane 20cm after dump plane with respect to 3.5cm before dump plane. As the premixed length increases homogeneity enhances in the mixture. With variation of distance the effect on homogeneity is dependent on the premixed length. FigureA11 shows the Mass-weighted Average Mass fraction of CH4 distribution along different planes with respect to time while figureA12 to 16 depict the Area-Weighted Average Velocity magnitude distribution for different amplitudes for W= 5, W= 20, W=35, W=50, W=80 and W= 95 respectively.

4. CONCLUSIONS

The gas turbine combustor development will have greater focus on both environmental pollution and energy conservation. Advanced concepts for environment pollution control required detailed understanding of the physical and chemical process occuring during combustion. Test data on laboratory Scale, bench scale, and prototype combustor are urgently needed for determining the details of the chemistry involved, fuel-air mixture, fuel injector design, combustor geometry, combustion air swirl, flow distribution and fuel spray characteristics. It should be mentioned in the velocity and mass fraction of CH_4 plots that mass fraction of CH_4

and velocity distribution may vary with number of iteration and that is the reason for the variastion of plots and figures. The variation of the mass fraction of methane and velocity fall into acceptable error range for numerical error, thus the results were accepted. The numerical simulation and the results qualitatively determine how the homogeneity of air-fuel mixture in a gas turbine combustor varied with variation of distance as well as time. Dump plane results are extremely important in this aspect as it signifies the first plane after the mixtures enter the main combustion zone. As the premixing length increases homogeneity enhances in the mixture. With variation of distance the effect on homogeneity depends on the premixing length.

5. Scope of future work

The dynamic characterization of mixing can be studied with different wave forms. Dynamic characteristic of combustion can also be studied. System identification foe mixing as well as combustion can be carried out.

REFERENCES

[1] A.H.Lefebvre, The Role of fuel preparation in Low-Emission Combustion, Journal of Engineering for Gas Turbines and Power-Transactions of the ASME, Vol.117, No.4, 1995, pp.617-554, http://dx.doi.org/10.1115/1.2815449.

[2] A.K.Gupta, Gas Turbine Combustion: Prospects and Challenges, Proc. International Symposium on Advanced Energy Conversion Systems and Related Technologies _ RAN95, Nagoya, JAPON, 1997, pp. 1311-1318.

[3] S.M.Correa, Power Generation and Aero Propulsion Gas Turbines, Proc. International Symposium on Combustion, From Combustion Science to combustion technology, 1998, pp 1793-1807.

[4] A.H.Lefebvre, Gas Turbine Combustion, Taylor & Francis, Philadelphia 1999.

[5] I.Glassman and R.A.Yetter, Combustion, Academic Press, UK 2008.

[6] J.R.Hertzberg, I.G.Shepherd, and L.Talbot, Vortex Shedding Behind Rod Stabilized Flames, Combustion and Flame, Vol.86, No.1-2, 1991, pp. 1-11, http:/dx.doi.org/10.1016/0010-2180(91)90051C.

[7] S.Chaudhuri and B.A.Cetegen, Blow off Characteristics of Bluff-Body Stabilized Conical Premixed Flames in a Duct with Upstream Spatial Mixture Gradients and Velocity Oscillations, Combustion and Flame, Vol.181, No.4, 2009, pp. 555-569, http:/dx.doi.org/ 10.1080/00102200802631278.

[8] A.Ateshkadi, V.G.Mcdonell and G.S.Samuelsen, Lean Blowout Model for a Spray-Fired swirl-Stabilized Combustion, Proceedings of the Combustion Institute, 2000, pp. 1281-1288.

[9] T.M.Muruganandam, S.Nair, D.Scarborough, Y.Neumeier, J.Jagoda, T.Lieuwen, J.Seitzman and B.Zinn, Active Control of Lean Blowout for Turbine Engine Combustors, Journal of Propulsion and Power, Vol.21, No.5, 2005, pp. 807-814.

[10] J.M.Cohen, J.H.Stufflebeam and W.Proscia, The Effect of Fuel/Air Mixing on Actuation Authority in an Active Combustion Instability Control System, Journal of Engineering for Gas Turbines and Power-Transactions of the ASME, Vol.123.No.3, 2001, , pp.537-542, http:/dx.doi.org/10.1115/1.1373399.

[11] Y.C.Chao, Y.L.Chang, C.Y.Wu and T.S.Cheng, An Experimental Investigation of the Blowout Process of a Jet Flame, Proceedings of the Combustion Institute, 2000, pp. 335-342.

[12] T.M.Muruganandam, B.H.Kim, R.Olsen, M.Patel, B.Roming, and J.Seitzman, Chemiluminescence Based Sensors forTurbineEngines, 39th, AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, Huntsville, Alabama, AIAA 2003-4490, 2003, pp 1-9.

[13] Ansys CFX ® v12.1, User Manual.

APPENDIX A

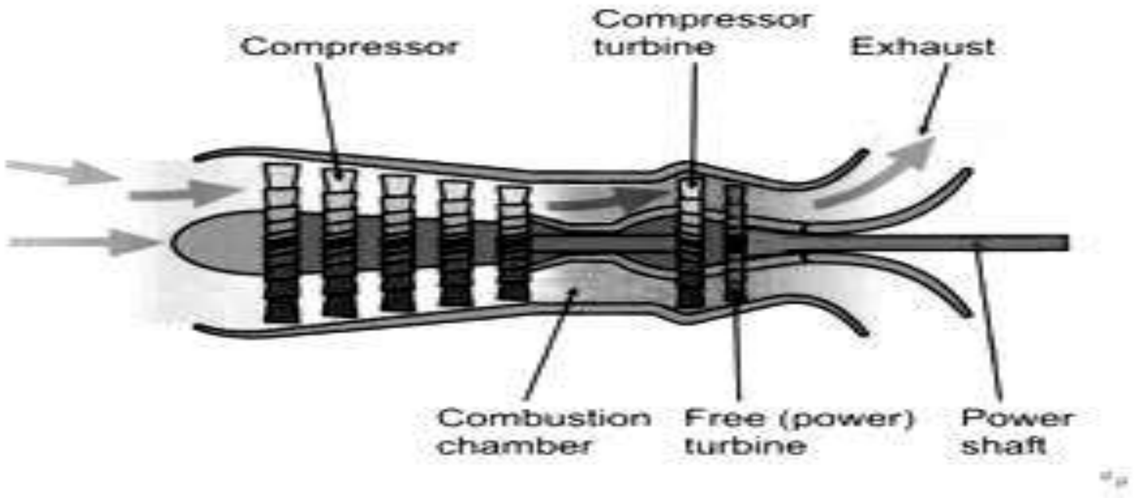

Figure A1.Schematic of gas turbine (P-1)

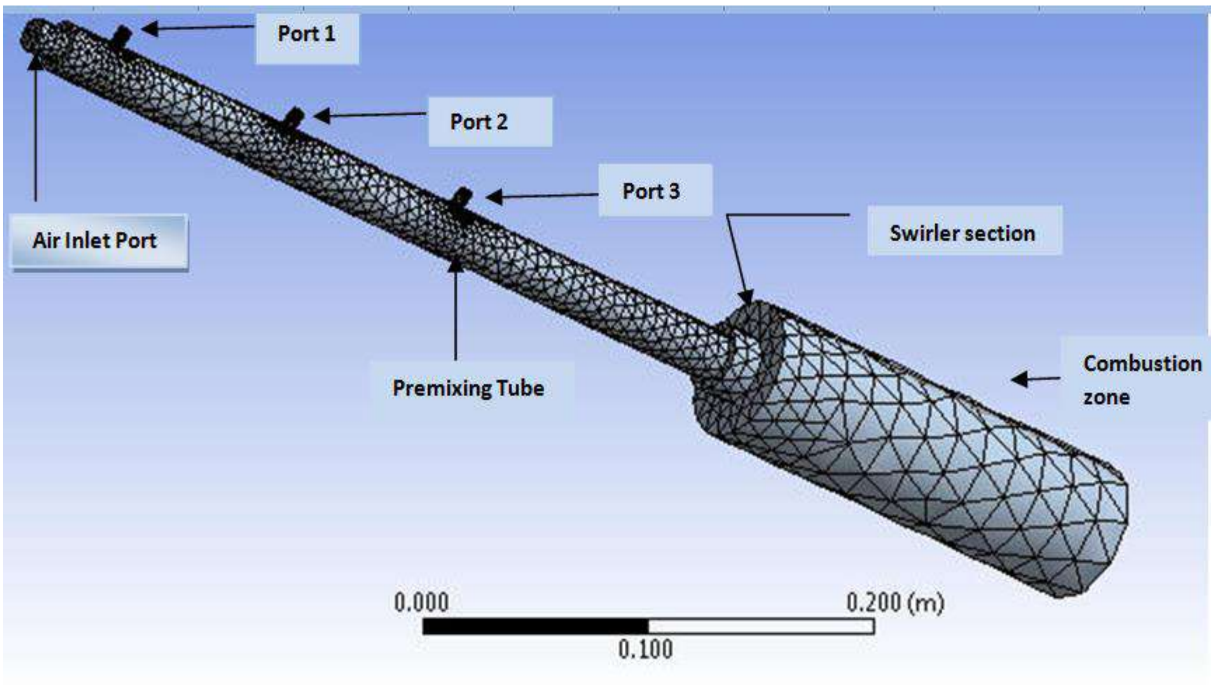

Figure A2.Meshed geometry used for ANSYS FLUENT simulation

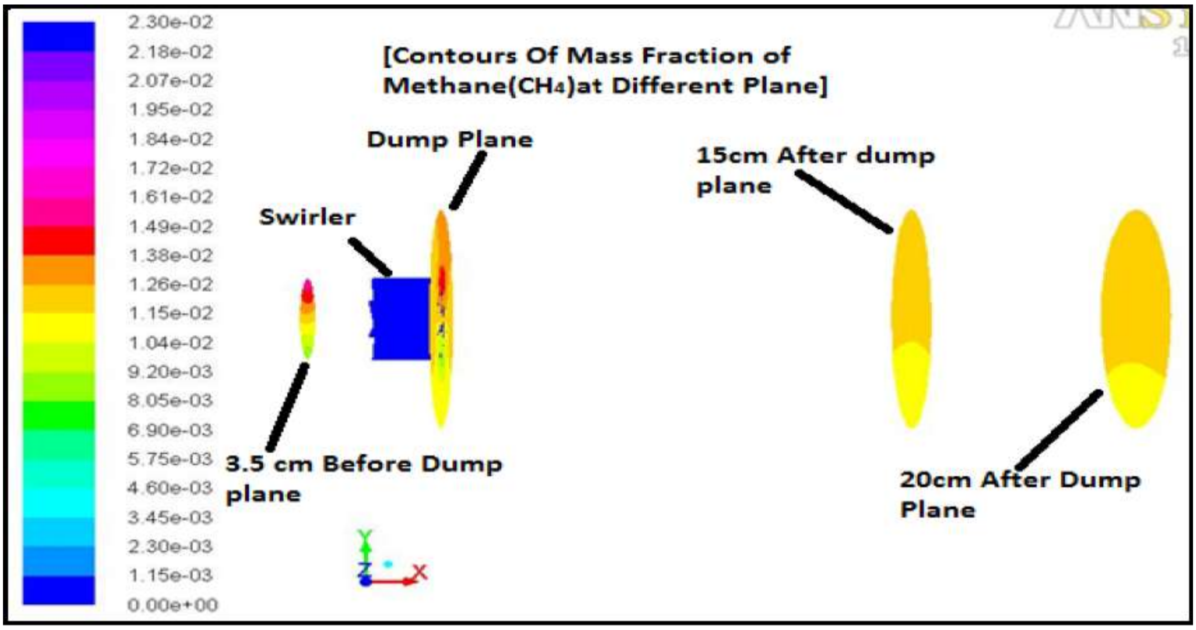

Figure A3.Contour of mass fraction of methane (CH4) at plane created before and after dump plane by using ANSYS FLUENT

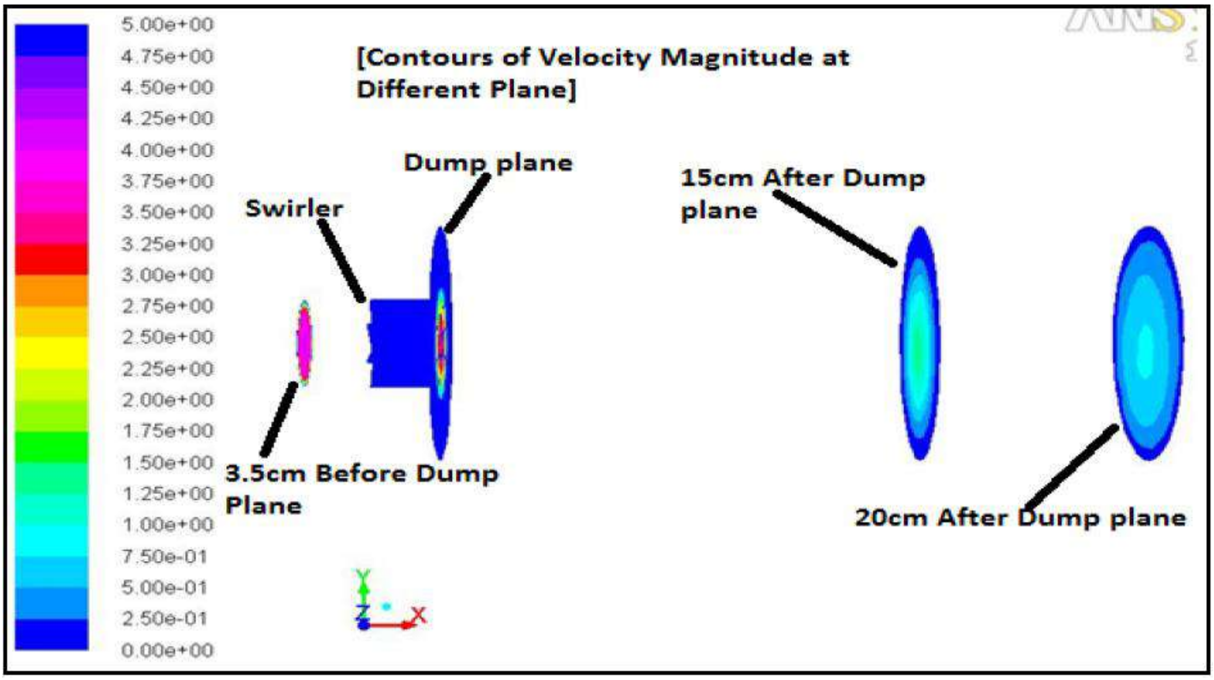

Figure A4.Contour of velocity magnitude at Plane created before and after dump plane by using ANSYS FLUENT

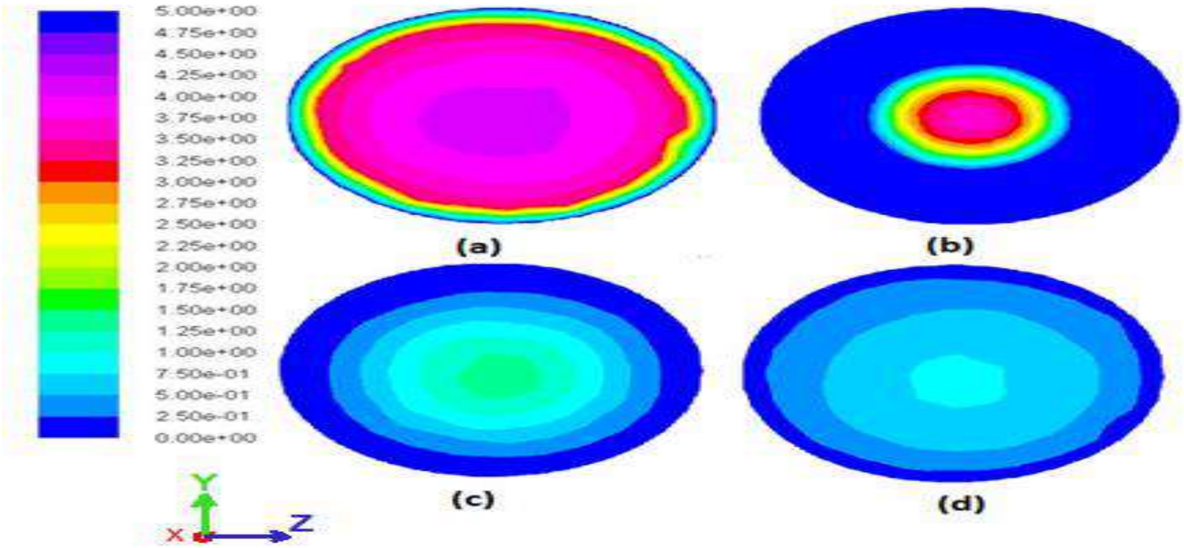

Figure A5.Velocity Magnitude Contour on Different plane for entry of fuel through port-1 for velocity V=1.062m/s and air flow Velocity of UDF (a)3.5 cm before dump plane (b)dump plane (c) 15cm after dump plane (d) 20cm after dump plane.

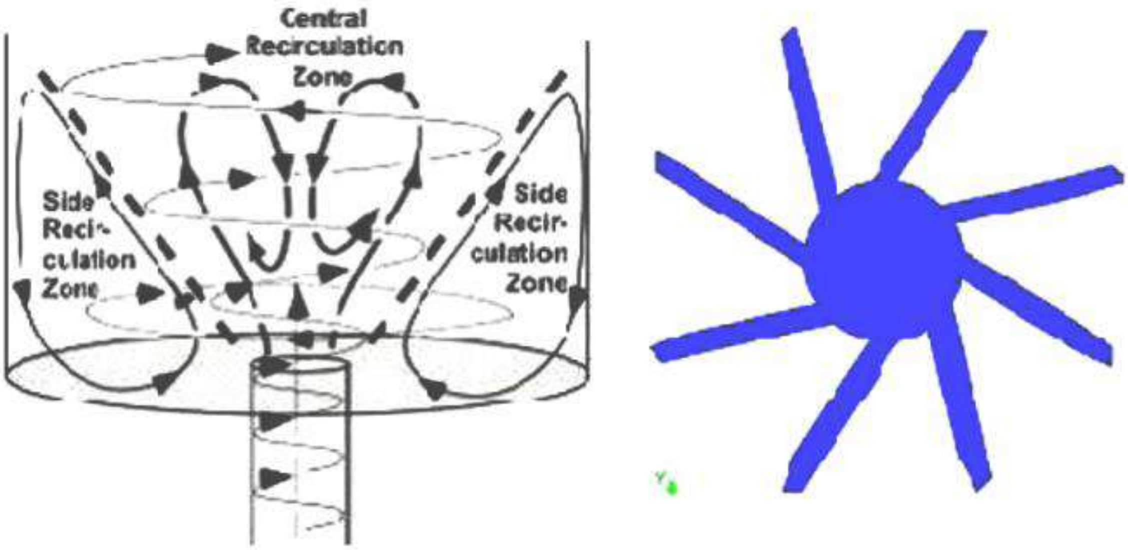

Figure A6.Recirculation zone and swirler

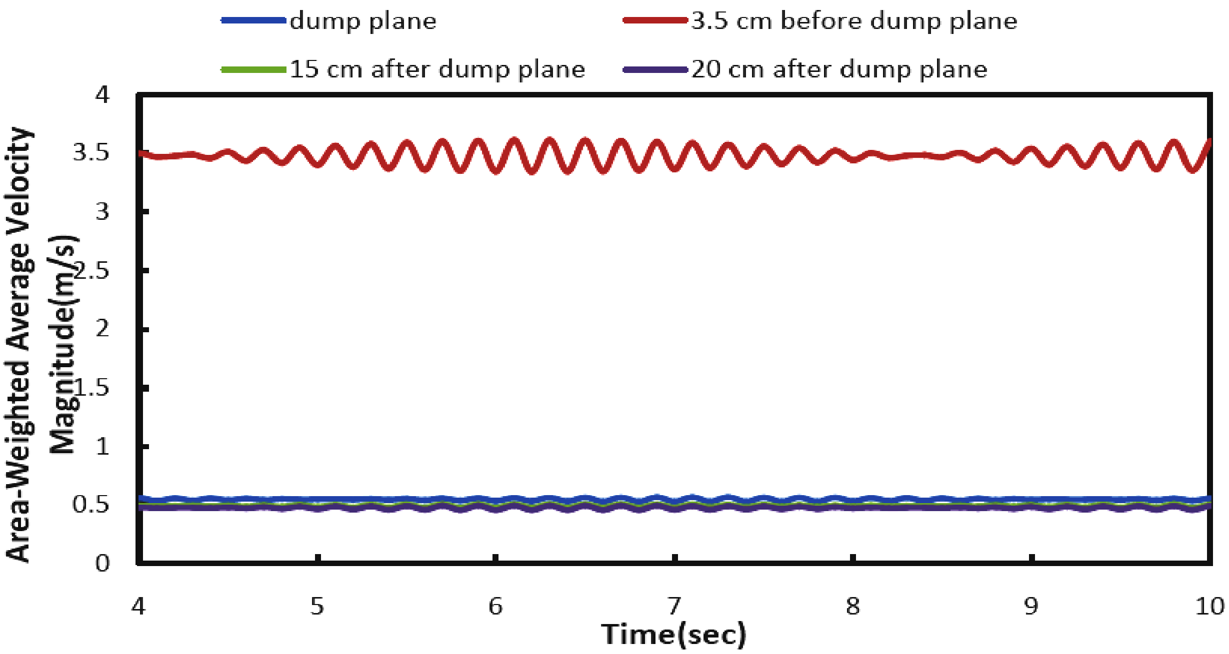

Figure A7.Area-weighted average velocity magnitude distribution along different planes with respect to time

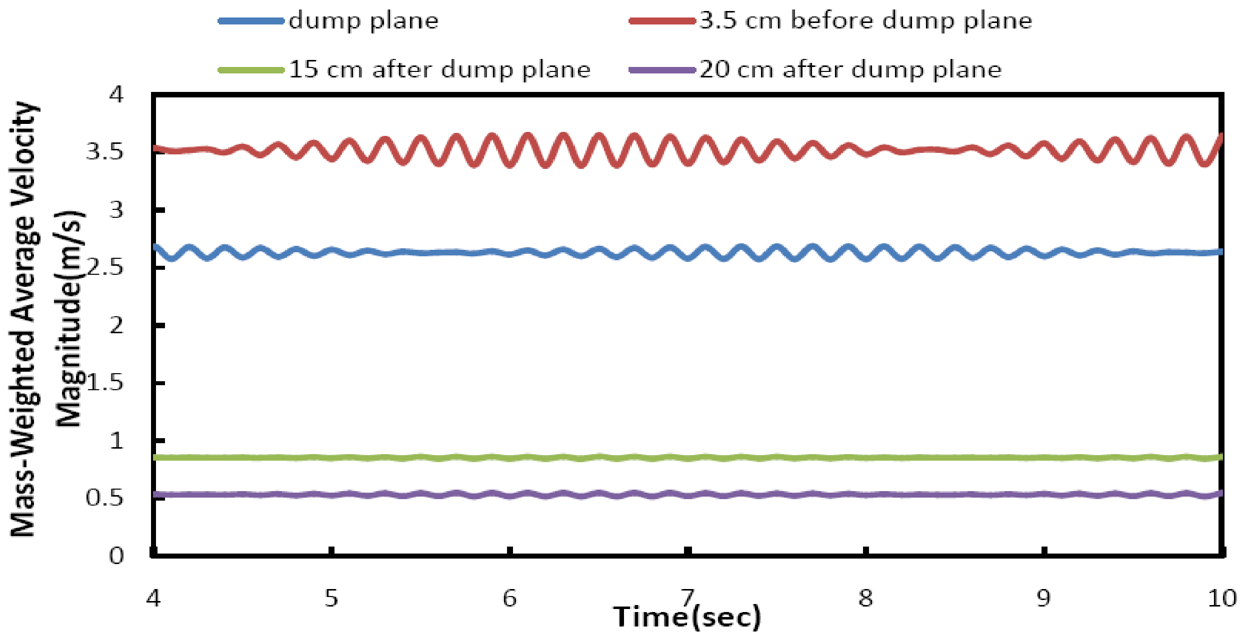

Figure A8.Mass-weighted average velocity magnitude distribution along different planes with respect to time

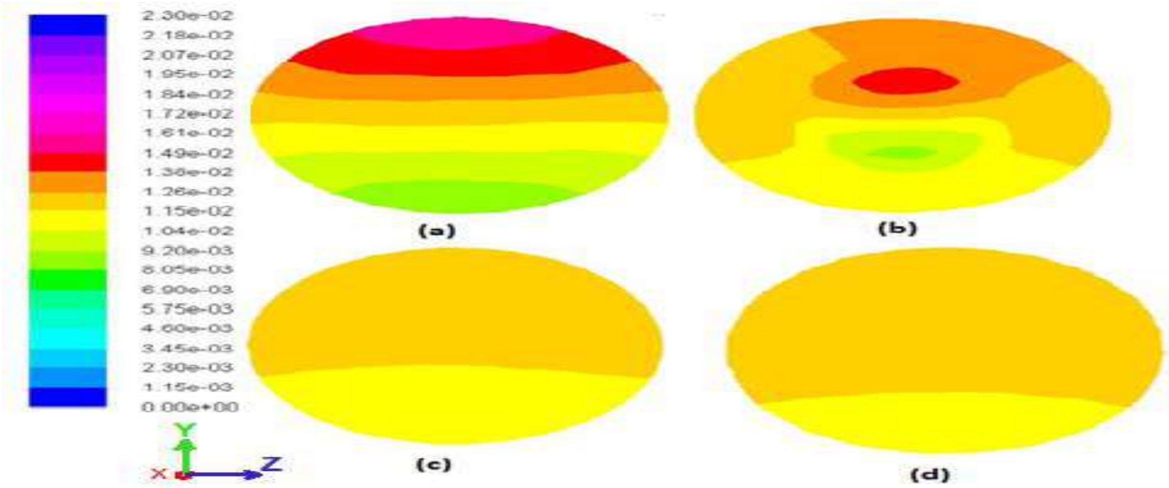

Figure A9.Mass fraction of CH_4 contour on different plane for entry of fuel through port-1 for velocity V=1.062m/s and air flow Velocity of UDF (a)3.5 cm before dump plane (b)dump plane (c) 15cm after dump plane (d) 20cm after dump plane

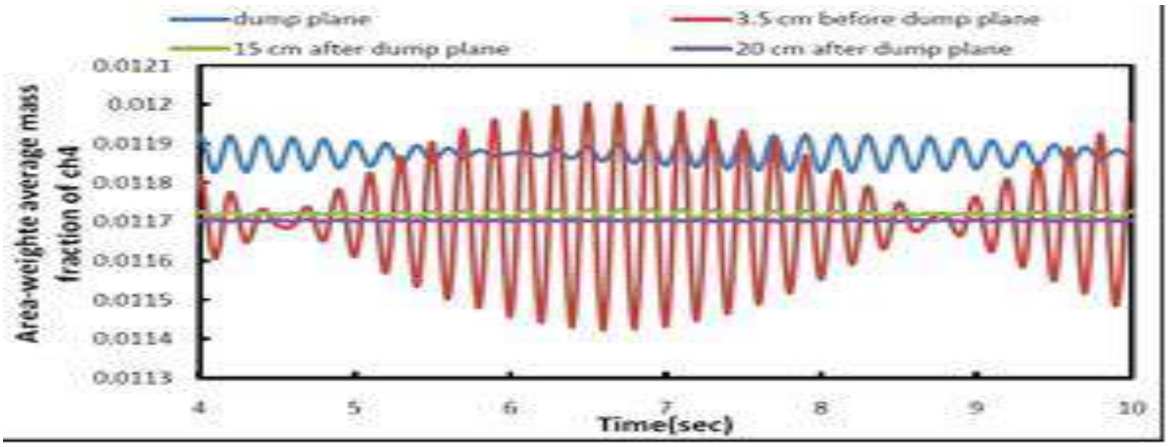

Figure A10.Area-weighted average mass fraction of CH4 distribution along different planes with respect to time

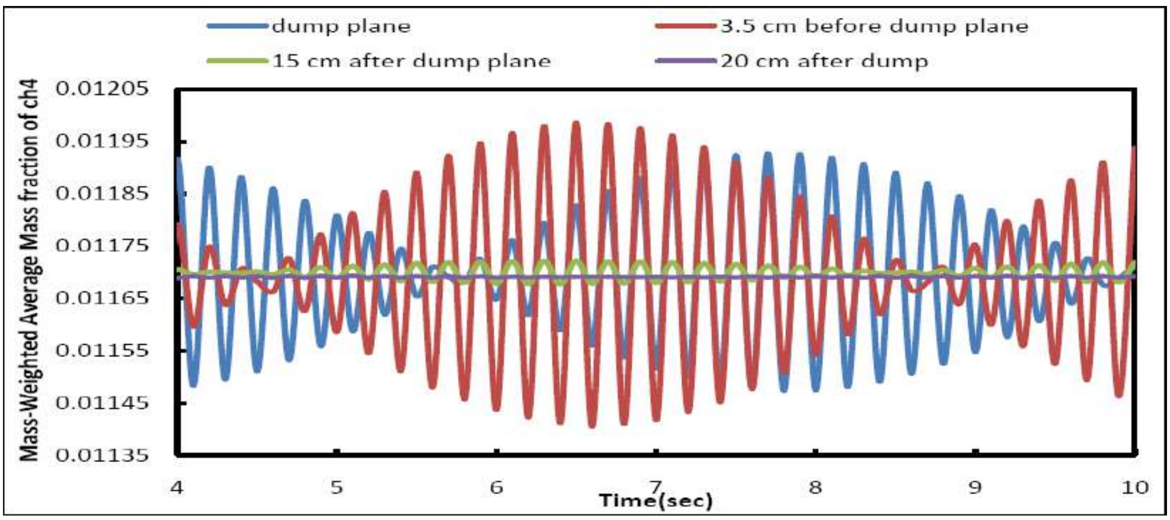

Figure A11.Mass-weighted Average Mass fraction of CH_4 distribution along different planes with respect to time

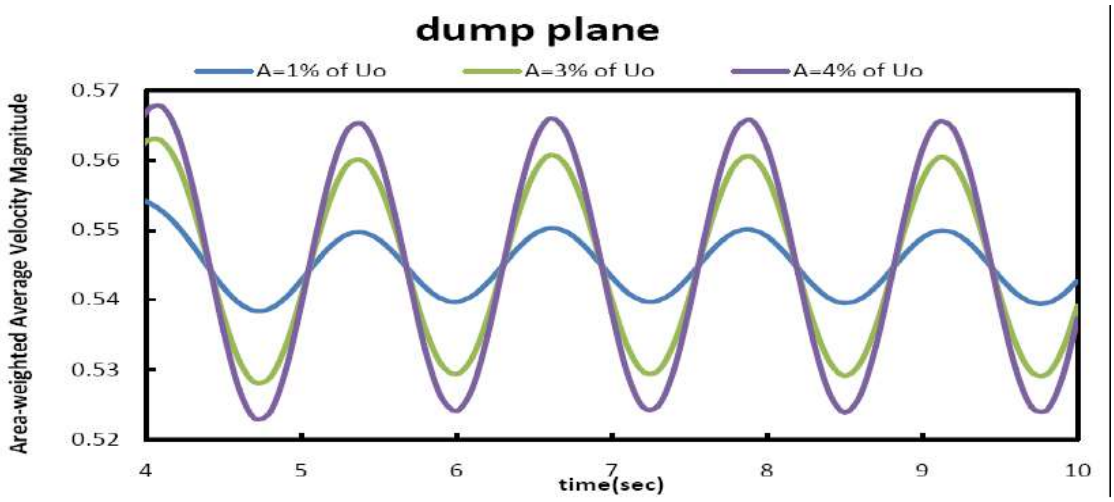

Figure A12.Area-weighted average velocity magnitude distribution for different amplitudes where $W = 5$

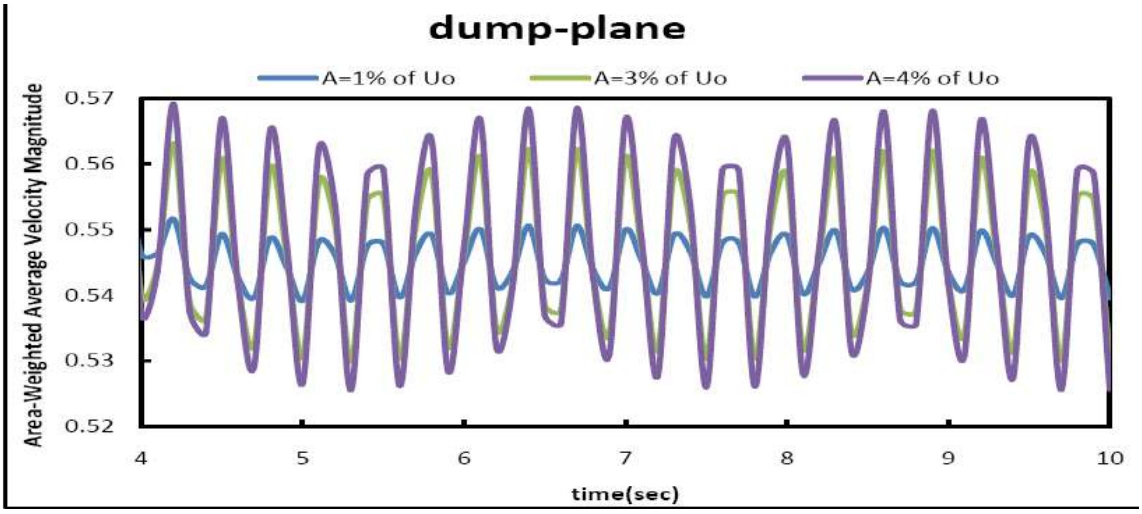

Figure A13.Area-weighted average velocity magnitude distribution for different amplitudes where $W = 20$

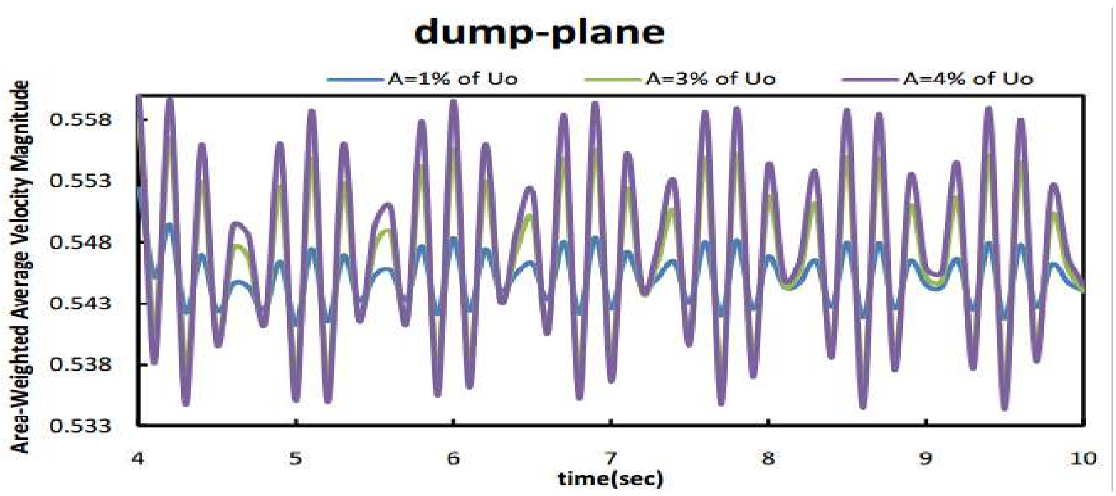

Figure A14.Area-weighted average velocity magnitude distribution for different amplitudes where $W = 35$

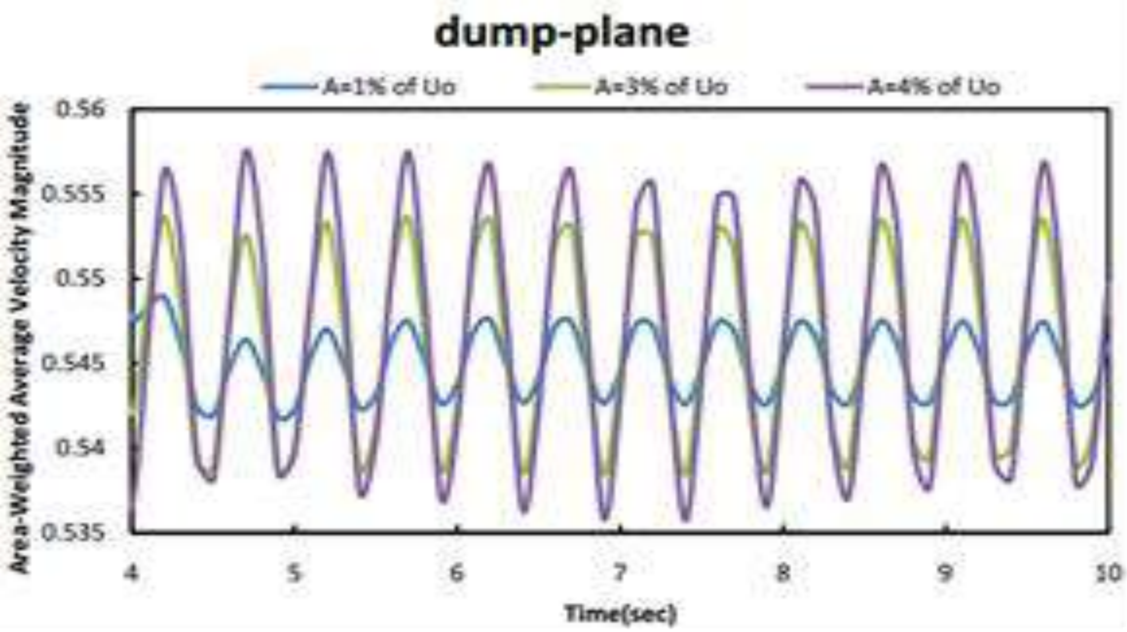

Figure A15.Area-weighted average velocity magnitude distribution for different amplitudes where W = 50

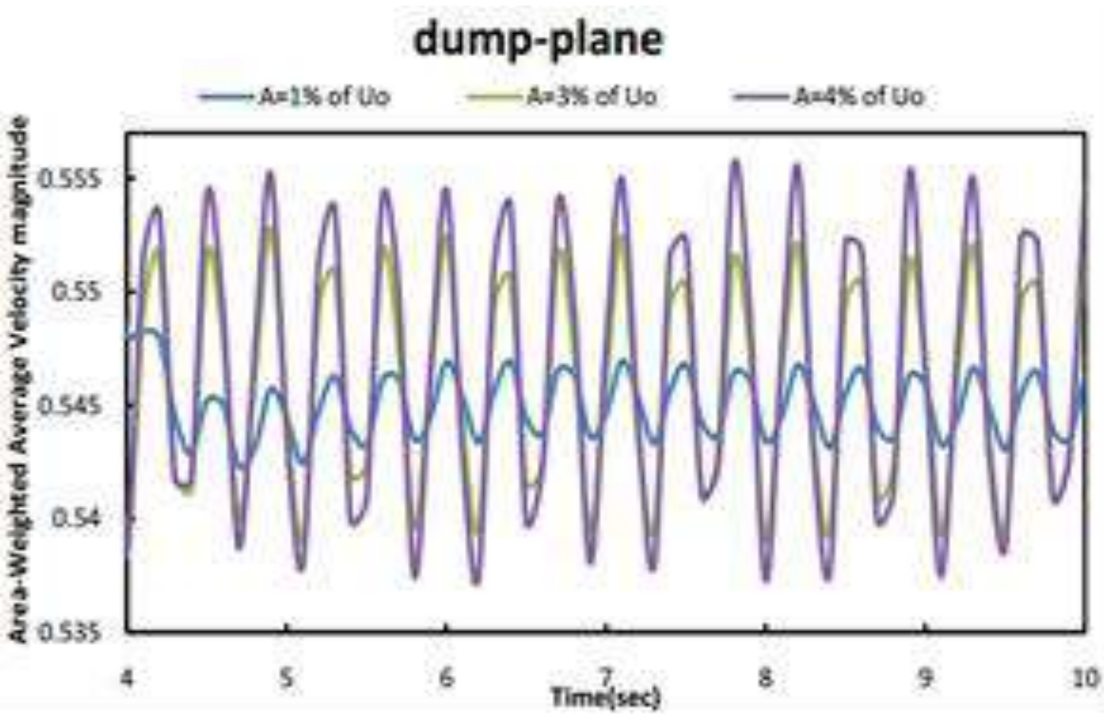

Figure A16.Area-weighted average velocity magnitude distribution for different amplitudes where W = 80